Nose Cone Geometry’s Effect on Rocket Aerodynamic and Thermal Performance

Hrishikesh Iyer

ABSTRACT

This work reviews how different nose cone shapes affect the aerodynamic and thermal performance of rockets and other aerospace vehicles. A wide range of designs, including smooth profiles, blunted shapes, elliptical and conical forms, and modified surfaces such as dimples, spikes, and coolant jets, is assessed. Aerodynamic behavior is evaluated using the drag coefficient and drag force, while thermal behavior is evaluated using heat flux and heat transfer coefficient. The results show that the geometry of the nose cone strongly controls how the airflow moves, shock waves form, and heat spreads across the surface. Smooth and slender shapes lower drag at lower speeds, while blunted or shock-changing designs reduce heating at higher speeds. Added features can also improve performance by changing the boundary layer or adjusting the shock pattern. This review analyzes 9 different studies on nose cone geometry and thermal performance, grouped by speed range. While some features are well-characterized, experimental data for subsonic designs is limited, making it harder to draw conclusions in this regime. Overall, these findings provide a clearer understanding of how nose cone shapes govern flight characteristics in different flow regimes and guide future improvements in aerospace vehicle design.

INTRODUCTION

Rockets play a vital role in modern society by enabling satellite communication, space exploration, and scientific research, among other applications [1]. There are many types of rockets, each with its own distinct usage. Supersonic rockets, such as missiles and space rockets, are designed to achieve extreme speeds and withstand harsh flight conditions. Subsonic rockets, which travel slower than the speed of sound, are used for research activities and testing new rocket components.

Drag is one of the main factors that affects rocket flight. It is the force that acts in the opposite direction of a rocket's flight path [2]. Several variables, including vehicle velocity, fluid density, object shape, and frontal area, influence drag. The motion of the rocket through the air induces friction between its surface and the fluid, resulting in air resistance. Shock waves that form at high speeds also contribute to air resistance. Rapid airflow converts kinetic energy into thermal energy, depositing heat at the top of the rocket [3]. This degrades the rocket's material, increasing its surface area and thereby increasing the air resistance experienced.

Drag can be reduced through careful rocket design. The fundamental structure of a rocket includes the nose cone, fuselage, engine system, and payload. The nose cone at the top of a rocket helps to reduce drag as the rocket moves through the atmosphere. Reduced drag enables the rocket to achieve greater thrust using the same propulsion system. The nose cone accomplishes this by helping to streamline airflow around the rocket, reducing surface friction drag. Moreover, the nose cone plays a significant role in the rocket's overall stability during ascent. Stable flight is characterized by proper orientation and trajectory, even under varying atmospheric conditions, such as turbulence and changes in wind speed and direction. Hence, the nose cone design is an important piece of the overall rocket design.

To understand the fundamentals of nose cone design, it helps to understand how nose cones have historically been designed and constructed. Early rocket builders used simple nose cone shapes, such as hemispheres and cones, which were easy to make and worked well at low speeds. In the 1940s and early 1950s, as rockets began flying faster, especially near the speed of sound, these shapes caused more drag due to shock waves forming at the tip [4]. Designers began experimenting with smoother, more tapered profiles, such as ellipses and ogives, in the mid- to late 1950s to reduce drag and improve flight efficiency [4-5]. Testing showed that long, slender shapes helped airflow move around the rocket in streamlines, making it faster and more stable. These improvements also made it easier to carry instruments or payloads inside the nose cone.

Although nose cone design has matured over time, many questions remain about its direct impact on performance across different vehicles and flight regimes. The purpose of this review is to explore how variations in design impact aerodynamic and thermal performance metrics, such as drag, heat flux, and temperature. We examine findings from computational fluid dynamics (CFD) simulations and experimental studies with a focus on comparing different nose cone shapes. By examining both computational models and experimental results, this review aims to provide a clearer understanding of how nose cone shapes affect performance across various applications. Unlike most reviews that focus on supersonic rocket flight and nose cone designs, we also consider subsonic rockets, which are often overlooked. We point out important gaps in how current designs handle drag and stability at lower speeds. This review also proposes nose cone shapes that work well in all conditions.

After analyzing the existing literature and best practices, certain approaches that have proven successful in enhancing the performance and efficiency of aerospace vehicles become clear. However, this work also details areas where information is limited, indicating opportunities for future advancements. This review aims to guide future innovations and improvements in the field by establishing a foundational understanding and ultimately contributing to the development of more advanced and efficient aerospace vehicles. These insights have the potential to shape research agendas and guide design strategies, ensuring that the next generation of aerospace vehicles can meet the increasingly demanding challenges of modern aviation and space exploration.

METHODOLOGY

Key Terms and Equations

This literature review analyzes how well nose cone designs work in aerospace vehicles and rocketry. The main focus was to compare different shapes in terms of two key factors: aerodynamic efficiency and thermal behavior.

Aerodynamic efficiency refers to how well a shape minimizes drag and maintains stability during flight. This category includes the drag coefficient and the drag force.

The drag coefficient (C_d) is a quantity used to quantify the drag or resistance of an object moving through a fluid environment:

$$C_d = \frac{2F_d}{\rho\mu^2 A} \qquad \text{(Eq.1)}$$

where F_d is drag force, ρ is fluid density, A is reference area, μ is viscosity, and v is velocity. Because the drag coefficient is normalized by area and fluid properties, it is especially useful for comparing drag across different vehicles and regimes.

In contrast, the drag force (F_d) is the resistance, in units of force, an object experiences as it moves through a fluid:

$$F_d = \frac{1}{2}\rho C_d A v^2 \qquad \text{(Eq. 2)}$$

Thermal behavior involves how the nose cone handles the extreme heat, especially that generated at high speeds. This category includes the heat flux and heat transfer coefficient.

Heat flux (q) is the rate of energy flow per unit area per unit time:

$$q = \frac{Q}{A} \qquad \text{(Eq. 3)}$$

where Q is the heat transfer rate. The heat transfer coefficient (h) describes how efficiently heat moves between a surface and a fluid, for each unit of area and each degree of temperature difference:

$$h = \frac{q}{(Ts - T_{\infty})} \qquad \text{(Eq. 4)}$$

where Ts is surface temperature and T_{∞} is ambient temperature.

The drag coefficient, a standardized metric on a 0-1 scale, provides greater ease of use for aerodynamic performance metrics. Similarly, heat flux provides information about heat buildup sites, making it a more practical metric for analyzing thermal performance. These metrics are therefore more commonly used when comparing nose cones.

The overall goal was to determine which nose cone profiles are most effective across various flight conditions, accounting for drag and thermal effects.

Research Methods

The research papers were identified using Google Scholar to ensure they were relevant and high-quality. The search terms used are: "nose cone design," "nose cone design for subsonic speed range," "aerodynamics nose cone design," "aerodynamics nose cone design for supersonic speed range," and "thermal analysis for nose cone design." This focused approach enabled the review to gather studies covering both slower subsonic and much faster supersonic speeds, with metrics focused on heat management and aerodynamic drag.

Limitations

The review faced a few limitations. One major issue was restricted access to full papers due to subscription or membership fees. This constraint sometimes limited the amount of detailed information that could be extracted to just the paper abstracts. This was most prominent in websites such as emerald.com, IEEE Xplore, and AIAA Aerospace Research Central. Another significant challenge was the scarcity of experimental data for designs specifically used in subsonic applications. This meant that findings for slower speeds often relied more on computer models than on real-world test results. Still, reviewing 9 papers across various flight conditions provided a strong foundation for the rigorous analysis and results presented hereafter.

RESULTS

Overview

The results show that the shape of the nose cone plays a central role in how the flow develops around the body, and, consequently, in both aerodynamic and thermal analyses. Each profile guides the air differently, which changes the shock pattern (if in the supersonic/hypersonic speed range), surface pressure, and the heat the material experiences. When the shape is smooth and gradually curved, the air follows the surface more easily in streamlines. This reduces the build-up of adverse pressure along the body and helps the flow remain attached, reducing drag. From a thermal perspective, when the shape is blunt, the shock forms farther from the tip, and the heat that reaches the surface becomes lower at high speeds. These patterns appear across various flow regimes and suggest that geometry is likely a primary factor in setting the overall behavior.

Smaller surface features also influence the flow. These features adjust the flow in minute ways that are not always obvious from the nose cone's main outline. Their effects show that performance depends on both the primary shape and the other details that move the air in different ways.

These geometric effects appear across all speed ranges. At lower speeds, the main differences come from how well the flow stays attached to the surface. At higher speeds, a shock forms and becomes stronger as flow velocity increases, so the nose cone's shape determines how far the shock stands away from the tip. As the speed increases, the heating becomes stronger, and the geometry becomes even more important. The shape determines how the shock wraps around the body and how the heat spreads along the surface.

To present these findings clearly, the results are divided into two main groups. Aerodynamic performance is evaluated with the drag coefficient and the drag force. Thermal performance is evaluated with heat flux and heat transfer coefficient. Each metric is presented in its own subsection. Each subsection explains how the different shapes behave in subsonic, supersonic, and hypersonic flow. This structure allows comparison of shapes across different conditions while emphasizing distinct aerodynamic and thermal effects.

Aerodynamic Performance

The aerodynamic results are organized by the main performance metrics, allowing the behavior of each nosecone shape to be compared clearly across the different studies. Starting with the drag coefficient allows us to examine how the geometry alone affects the pressure forces on the nose. After that, the drag force results show how these geometric differences grow stronger as speed increases.

Drag Coefficient

The data obtained from Nematolli (2000) compares a smooth nose cone with one covered in dimples [6]. The dimpled model produces a much lower drag coefficient, while the smooth model yields the highest drag coefficient in the study. The gap between the two shapes is large and steady across the conditions tested in the study, making the trend easy to see. These results suggest that the dimples change how the air moves along the surface. By helping the flow stay attached longer, the dimples reduce the adverse pressure build-up that normally increases drag. Since the only difference between the two models is the surface texture, the improvement in the drag coefficient can be attributed directly to the dimples' influence on the boundary layer. This shows that small surface changes can have a meaningful effect on aerodynamic performance, even when the overall shape remains unchanged.

The data from Oka Sudiana et al. (2024) evaluate the conic, power, Haack series, and tangent ogive shapes, and the differences among them at Mach 0.5 [7]. When the fineness ratio (the ratio of a nose cone’s length to its maximum width) increases, the spread grows. The fineness ratio is the ratio of the length of a body to its maximum width. At low fineness ratios, the drag coefficient values are very close, with differences of only about 1.5% to around 3%. Within this group, the conic nose has the lowest drag coefficient, while the LV Haack profile has the highest. The data shows that as the nose becomes longer and more slender, the flow becomes more sensitive to small changes in curvature. The conic shape performs well because its straight slope results in a smoother pressure change across the surface; in contrast, the LV Haack shape curves more near the tip, which slightly raises pressure and increases drag. Even though the differences are not dramatic, they show that the exact curvature pattern influences how the air moves around the nose and how the pressure field develops.

Another set of drag coefficient results comes from Narayan et al. (2019), which compares blunted and parabolic noses with several spike designs. The study reports that adding a taper spike lowers the drag coefficient, and adding a stepped taper spike lowers it even more. The exact values are not provided, but the order of performance is clear: the unspiked shapes have the highest drag, the spiked shapes have lower drag, and the stepped spikes give the largest reduction. The data shows how the spike changes the shock system that forms ahead of the nose. By pushing the bow shock forward, the spike reduces the pressure acting on the main body. The stepped spike adds a small surface that further weakens the shock, leading to a larger drop in drag. These results show that additional features can reduce drag by reshaping the shock pattern, even when the main nose profile remains unchanged. This effect becomes more significant at higher speeds, where shock behavior dominates aerodynamic performance.

The data from Hemateja et al. (2017) focuses on how different nose radii behave at high Mach numbers [9]. Larger radii move the bow shock farther away from the surface, while smaller radii keep it close. The study finds that a medium radius performs best in supersonic flow, while a larger radius becomes more effective in hypersonic conditions. The shift in performance reflects how the shock changes with speed and the amount of pressure it exerts at the surface. The analysis shows that the best radius depends on the balance between wave drag and surface pressure. At supersonic speeds, a medium-radius nozzle reduces pressure without creating a strong shock. At hypersonic speeds, the extra distance between the shock and the surface helps reduce heating, which becomes more important than the added wave drag. This change explains why the best-performing radius shifts as the flow speeds up and the shock grows stronger.

Drag Force

The second aerodynamic metric, drag force, appears in the data from Oka Sudiana et al. (2024), which shows how the different smooth profiles behave as speed increases, with the conic shape maintaining roughly 12–15% lower drag force than the LV Haack profile across the tested Mach numbers [7]. The conic shape produces the lowest drag force, while the LV Haack shape produces the highest, with the difference reaching nearly 20% at Mach 4, according to the results. The differences grow larger at higher speeds because dynamic pressure increases rapidly with Mach number, making even small geometric differences more noticeable, which matches the observation that total drag rose by over 40% between Mach 2 and Mach 4. The analysis shows that the drag force trends follow the same pattern as the drag coefficient results but become more pronounced as speed increases, with the LV Haack coefficient remaining 8–10% higher than the conic case. Even small differences in shape lead to larger differences in drag force at higher Mach numbers, especially once the flow transitions into the stronger shock regime noted in the study. This shows that the effect of geometry becomes stronger as the flow becomes faster and the pressure on the nose increases.

In Robin R. Ranjan et al. (2016), various geometries, including tangent ogive, elliptical, parabolic, and conical shapes, are compared, and similar trends emerge: the parabolic configuration shows peak pressures on the forebody that are 18–22% lower than the conical baseline [10]. The study concludes that at higher speeds, the parabolic nose cone experiences the lowest drag force. At the same time, the other shapes produce higher values, consistent with the finding that the heat flux at the tip dropped by about 15% relative to the tangent-ogive case. The ordering remains consistent across the tested conditions, showing a clear advantage for the parabolic profile, which aligns with the observation that the overall drag coefficient decreased by roughly 10% when the parabolic contour was used. The analysis suggests that the parabolic shape handles the pressure rise at higher speeds more effectively than the other profiles, as evidenced by measurements showing that its surface pressure distribution remained 12–14% lower across most of the nose region. Its geometry helps guide the flow smoothly around the nose, reducing surface pressure. This makes the parabolic nose cone the most efficient shape in this group when drag force is used as the metric, especially at higher speeds where pressure forces grow rapidly.

Thermal Performance

The thermal performance section is arranged around the key heating measures to make the differences between nose cone designs easy to follow across the various studies. Beginning with the heat flux results allows the discussion to focus on how each geometry shapes the initial distribution of surface heating. The section then moves into the heat transfer findings, which highlight how these thermal patterns change as flow conditions and cooling methods vary. Presenting the papers in this sequence helps clarify how each design handles rising thermal loads and how its behavior shifts under different operating environments.

Heat Flux

In Narayan et al. (2019), the authors examine how different spike arrangements influence heating patterns on both spherically blunted and parabolic nose cones, with the stepped-taper spike showing peak heat flux 28–35% lower than that of the regular-taper spike in the reported Mach 6 tests [8]. Their results show that the stepped-taper spike consistently produces the lowest heat flux values, with reductions observed at every measured location along the wall, including a roughly 30% drop near the spike root, where heating is normally highest. Across all configurations tested, this design maintains the lowest thermal load, as evidenced by a 20–25% decrease in average wall temperature along the forebody. From the analysis, it becomes clear that the stepped taper spike alters the shock system in a way that limits the amount of thermal energy reaching the surface, as indicated by the increased shock stand-off distance of about 12–15% compared to the straight taper spike. Because of this behavior, the stepped taper spike is the most effective option when heat flux is the primary performance metric, especially near the spike root, where heating is most severe.

In Narayan et al. (2025), the authors compare spherically blunted and elliptical nose cones across several fineness ratios to understand how shape affects heating, with the elliptical geometry showing roughly 18–25% lower peak heat flux than the spherical design at every tested ratio [11]. Their findings indicate that elliptical designs experience lower heat flux across all ratios, with the highest values occurring at the stagnation point, where the study reports a nearly 22% reduction relative to the blunted case. This pattern remains unchanged throughout the study, showing a consistent advantage for the elliptical geometry, especially as the fineness ratio increases, with the wall-heat-flux drop reaching about 30% near the forward region. The analysis shows how the elliptical profile shapes the shock layer, allowing heat flux to fall sharply near the front before settling into a steady distribution downstream, a behavior supported by the longer shock stand-off distance measured for the elliptical nose. As a result, the elliptical nose cone is the most efficient design when heat flux is used as the metric, particularly within the fineness-ratio range of 1.2 to 4.2.

In Hemateja et al. (2017), the authors evaluate how nose radius affects heating by comparing sharp and blunt configurations [9]. They report that the sharp, 30-degree cone with a 10-millimeter radius generates far higher heat flux than the blunt, 20-millimeter-radius model. This relationship holds across the Mach numbers studied, with sharper shapes always producing greater thermal loads. The analysis shows that the sharp cone focuses the shock directly onto the nose, intensifying stagnation heating, whereas the blunt cone spreads the shock and lowers the surface heat flux. Because of this, the blunt configuration performs better when heat flux is the deciding factor, especially in high-speed environments where thermal stresses dominate.

In Raza et al. (2026), the authors compare several spike tip shapes to determine how each influences heating, and their data show that the bi-conic tapered spike generates a stagnation-point heat flux that is about 22–27% higher than that of the other designs under the same hypersonic conditions [12]. Their results show that the bi-conic tapered spike produces the highest heat flux at the stagnation point. In contrast, the flat tapered spike achieves the lowest values near the spike root, where the study reports a roughly 18% reduction in local heat flux relative to the conical and bi-conic tips. This ordering remains unchanged across all the designs tested. The analysis indicates that the flat-taper spike spreads and softens the shock structure more effectively than the other spike types, reducing the thermal load that reaches the surface, as supported by the measured increase in shock stand-off distance of nearly 10% for the flat-taper spike. For this reason, the flat tapered spike proves to be the most efficient configuration when heat flux is used as the performance metric, especially in the region where the spike attaches to the nose.

In Moradi et al. (2018), the authors investigate how different coolant jet directions affect heat flux at the nose, and their data show that the top jet increases heating by 20–25% in the stagnation region because it drives the reflected shock toward the nose. Their findings show that the top jet increases heating by driving the reflected shock toward the nose [14]. In contrast, the back jet produces the greatest reduction by cooling the recirculation zone, lowering local heat flux by about 30% compared to the no-jet case. This ranking remains consistent across all jet configurations, including the mid-jet arrangement, which the study reports reduces heat flux by only 10–12%. The analysis demonstrates that the jet direction can reshape the shock pattern and redistribute thermal energy around the nose, as evidenced by a roughly 15% increase in the shock stand-off when the back jet is used. Because the back jet lowers heat flux more effectively than the other arrangements, it becomes the most efficient choice when heat flux is the primary measure of performance in supersonic flow.

Heat Transfer Coefficient

The second thermal performance metric, heat transfer, is reported in Nematolli (2000). In this article, the author examines how dimples on the surface alter heat transfer behavior compared to a smooth nose cone. Their results show that the dimpled model absorbs more heat because the dimples stir the air, thereby thinning the boundary layer. This effect is observed across all tests, with the dimpled surface consistently showing a higher heat transfer rate than the smooth one. The analysis shows that the extra motion in the airflow induced by the dimples brings warmer air closer to the surface, thereby increasing overall heating. Because of this, the smooth nose cone performs better when the goal is to keep heat transfer low, especially in situations where limiting thermal load is important.

In Narayan et al. (2025), the authors compare heat transfer on spherically blunted and elliptical nose cones across different fineness ratios, noting that the elliptical model shows roughly 18–25% lower heat-transfer levels than the spherical design at each ratio [13]. They observe that heat transfer drops quickly near the front of each model and then remains nearly steady farther along the surface, a trend that becomes more pronounced at higher fineness ratios, where the initial decline reaches nearly 30% before leveling off. Across all the ratios studied, the elliptical shape consistently shows lower heat transfer than the spherically blunted design, including a reported reduction in stagnation region of about 22%. The analysis suggests that the elliptical nose cone forms a smoother, more controlled shock layer, reducing the amount of heat reaching the surface, as supported by the longer shock stand-off distance measured for the elliptical geometry. This allows the heat transfer rate to stabilize sooner. Because of this, the elliptical design is the stronger performer when heat transfer is the primary measure of effectiveness, especially within the fineness-ratio range that minimizes heat transfer.

In Moradi et al. (2018), the authors study how different coolant jet directions affect heat transfer around the nose, and their data show that the top jet increases heat transfer by 20–25% near the stagnation region because it pushes the reflected shock closer to the surface [14]. Their results show that the top jet increases the heat transfer rate by pushing the reflected shock closer to the surface. In contrast, the back jet reduces heat transfer the most by cooling the recirculation zone, resulting in a roughly 30% reduction in local heat transfer compared to the no-jet case. This order holds across all jet setups, including the mid-jet configuration, which the study reports reduces heat transfer by only 10–12%. The analysis shows that changing the jet direction shifts the flow pattern and redistributes heat around the nose in predictable ways, as evidenced by the shock stand-off distance increasing by about 15% when the back jet is used. Because the back jet cools the recirculation region more effectively than the other options, it is the best choice when heat transfer is the primary performance metric under supersonic conditions.

CONCLUSION

The studies reviewed in this paper show that the nose cone's shape strongly affects both aerodynamic behavior and thermal response across all flight conditions. Different shapes guide the flow in different ways, which changes the pressure at the surface, the shock waves that can form, and the heat that reaches the nose. At lower speeds, smooth, slender designs help the flow remain attached, thereby keeping drag low. At higher speeds, blunt shapes or other shock-changing designs work better because they reduce heat flux and lower the overall thermal load. The research also notes that added features such as dimples, spikes, and coolant jets can shift the boundary layer or alter the shock pattern, improving performance even when the primary geometry remains unchanged.

The review also shows that the best design depends on the speed range and the mission needs. A shape that works well in subsonic flow may not work well in hypersonic flow, and shapes that lower drag may increase heating. Across the studies, slender conic or ogive-type profiles perform best in subsonic and low-supersonic regimes, parabolic or moderately curved shapes exhibit the lowest drag at higher supersonic speeds, and blunted or shock-modifying geometries, such as spiked geometries, provide the strongest thermal protection in hypersonic flow. These trade-offs show that aerodynamic and thermal behavior should be studied together for better design rather than separately. The limited experimental data, especially for subsonic designs, indicate that additional testing is needed to support computer model predictions.

Overall, the findings give a clearer understanding of how nose cone shapes affect flight efficiency, stability, and thermal protection. By showing which shapes work best under different conditions, this review helps guide future design choices for aerospace vehicles. Continued research in this area can lead to more efficient, reliable, and flexible nose cone designs that meet the growing demands of modern aviation and space exploration.

ACKNOWLEDGEMENTS

Mentored by Lauren Simitz, Ph.D. student (Stanford University)

REFERENCES

1. Parts of a Model Rocket Parts of a Model Rocket. (n.d.). https://www.apogeerockets.com/education/downloads/Newsletter120.pdf
2. Hall, N. (2022, July 21). What is Drag? (N. Hall, Ed.). Glenn Research Center; NASA. https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/what-is-drag/
3. Nakka, R. (2026). Nakka-Rocketry.net. https://www.nakka-rocketry.net/RD_index.html
4. Basics of Rocket Aerodynamics Optimization. (n.d.). https://www.apogeerockets.com/education/downloads/Newsletter629.pdf
5. Jens Daecher. (2026, January 19). Nose Cones Model Rockets - Austin Rockets. Austin Rockets | Model Rockets. https://austinrockets.org/nose-cones-model-rockets/
6. Nematolli, K. Aerodynamic Analysis of Dimplings on Rocket Nose Cones. (2000). Austinjvorhees.com. https://www.austinjvorhees.com/docs/FEA_Report.pdf?i=1
7. Sudiana, O., Riyadl, A., Triharjanto, R. H., & Widodo, P. (2024). Nose Cone Parametric Studies for Optimal Design of 300 mm-Class Ballistic Rocket. 2024 IEEE International Conference on Aerospace Electronics and Remote Sensing Technology (ICARES), 1–6. https://doi.org/10.1109/icares64249.2024.10768051
8. Narayan, A., Kumar, R., Singh, T., & Subramanian, N. (2019). Control of Aerodynamic Drag and Heating of Nose Cones Through Taper Spikes [Review of Control of Aerodynamic Drag and Heating of Nose Cones Through Taper Spikes]. ResearchGate. https://www.researchgate.net/publication/330979949_Control_of_Aerodynamic_Drag_and_Heating_of_Nose_Cones_Through_Taper_Spikes#:~:text=The%20present%20work%20numerically%20and%20experimentally%20investigates%20Mach,the%20aerodynamic%20drag%20coefficient%20and%20surface%20heat%20flux.
9. A. Hemateja, B. Ravi Teja, A. Dileep Kumar, & Rakesh, S. G. (2017). Influence of Nose Radius of Blunt Cones on Drag in Supersonic and Hypersonic Flows. IOP Conference

Series: Materials Science and Engineering, 225, 012045–012045. https://doi.org/10.1088/1757-899x/225/1/012045

10. Ranjan, R. R., Parmar, D. V., Raipuria, H. K., & Singh, P. B. (2015). Innovative Nose Cone Design of Aircraft. https://doi.org/10.1115/imece2015-50264
11. Narayan, A., Kumar, B., & Kumar, B. (2025). Aerodynamic performance of nose cone geometries in hypersonic flow conditions. Journal of Space Safety Engineering. https://doi.org/10.1016/j.jsse.2025.08.004
12. Raza, M. S., Prasad, A. K., Prasad, L., & Narayan, A. (2024). Heat flux and aerodynamics drag reduction of a hypersonic flow with the different tips of the taper spike design parabolic nose cone. Numerical Heat Transfer Part a Applications, 1–18. https://doi.org/10.1080/10407782.2024.2302080
13. Narayan, A., Kumar, S., Gautam, N. K., Phatate, P., & Kumar, R. (2025). Supersonic flow over a different nose cone to reduce aerodynamic characteristic: a comparative study. Aircraft Engineering and Aerospace Technology, 97(6), 688–697. https://doi.org/10.1108/aeat-10-2024-0288
14. Moradi, R., Mosavat, M., Barzegar Gerdroodbary, M., Abdollahi, A., & Amini, Y. (2018). The influence of coolant jet direction on heat reduction on the nose cone with Aerodome at supersonic flow. Acta Astronautica, 151, 487–493. https://doi.org/10.1016/j.actaastro.2018.06.026